International Journal of Mechanical and Production Engineering Research and Development (IJMPERD)
ISSN(P): 2249-6890; ISSN(E): 2249-8001
Vol. 9, Issue 4, Aug 2019, 643-648

Original Article

FINITE ELEMENT ANALYSIS OF CENTER PIN AND BRACKET OF JIG – FIXTURE ASSEMBLY

RAJWARDHAN KULTHE[1], VIRAJ MEKHE[2], MIHIR MORE[3]
SANGRAM NANDAWADEKAR[4] & VIKAS DEULGAONKAR[5]

[1,2,3,4]Department of Mechanical Engineering, Marathwada Mitra Mandal's,
College of Engineering, Pune, Maharashtra, India

[5]Dean & Associate Professor, Department of Mechanical Engineering,
Marathwada Mitra Mandal's, College of Engineering, Pune, Maharashtra, India

ABSTRACT

The manufacturing industry caters the range of products to satisfy the ever changing market needs. To overcome the increasing production demands, the industry implies various techniques. We need a technique for increasing the production of drilling two holes on a Railway Pinion, improve the quality of product and reduce the operation time. This project aims to design the Jig and Fixture for the same. The 3-dimensional Computer Aided Model of the components is made using CATIA V5-R21 software. To study the behavior of component, simulation is carried out. Preprocessing of the CAD model is carried in Hypermesh software. Boundary conditions are applied using physical situations of the components. Finite element analysis of the components is done, and the results obtained are compared with the theoretical analysis and also with the available literature. The stresses and deformations are found within desired limits. Using FE analysis, the parts are manufactured and assembled.

KEYWORDS: ***Design, Drill Jig, Manufacturing, Stress, Deformation & Static Analysis***

Received: May 10, 2019; **Accepted:** May 30, 2019; **Published:** Jul 06, 2019; **Paper Id.:** IJMPERDAUG201964

INTRODUCTION

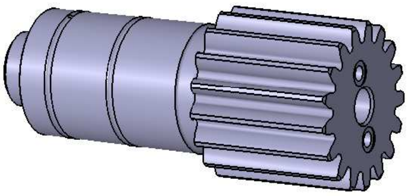
For any manufacturing unit, productivity is of great concern to maintain the profitability and sustainability of the unit. Various factors govern productivity of the manufacturing unit, which includes but not limited to skilled man-power, quality of products, availability of machinery and raw material i.e. inventory, human factor, working environment and many more. However, with the availability of modern machinery, the heart of production unit has become the design of Jig and Fixtures. These are the sole components, which keep on changing the available machinery as per product dimensions and shape. One such case is considered in present the work. The objective is to minimize the cycle time and simplify the drilling operation for Radial drilling machine. This is done using classical/theoretical approach. Present literature reveals the use of CAD/CAE and experimentation techniques to validate the classical design.

Design and Analysis of indexing jig for drilling operation discussed by NBV Lakshmi Kumari and G. Prasanna Kumar [II] mainly focuses on designing aindexing Jig and analysis backed up by manual calculations. Finite element analysis and experimental simulation of Chassis Mounted Platform for off-road wheeled combat and transport utility vehicles are discussed by Deulgaonkar V. R [III] while dealing with designing of the mounting

platform along with its analysis for stress and deflection present in the said component.

WORK-PIECE DESCRIPTION

The work-piece considered under the project is the Pinion used in Indian Railways' transmission system, which is press fitted in the tapered hole. After due time, the pinion needs to be taken out for timely maintenance. The Gun drills are used to pass pressurized fluid in the tapered section. This function allows the pinion to be easily removed from the system.



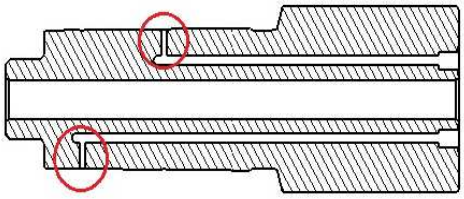
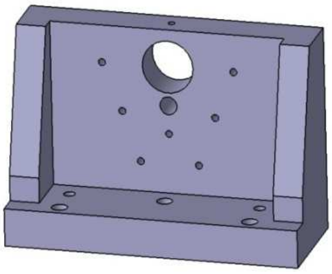


Figure 1: Pinion Work-Piece **Figure 2: Cross-Sectional View of Pinion**

The main motive of the project is to design, analyze and manufacture a 'Jig and Fixture' for drilling two holes on the circumference for proper flow of lubricant in the gear.

CAD MODELING

In present work, considering the length to diameter ratios for circular components and length to respective dimension ratio for non-circular components, solid modeling technique is used. To avoid complexity during finite element analysis, the computer aided model is prepared using extrude, pocket, pad, hole and chamfer commands. For Computer aided design of the components, use of CATIA V5-R21 is done. The computer aided models are shown in figure.



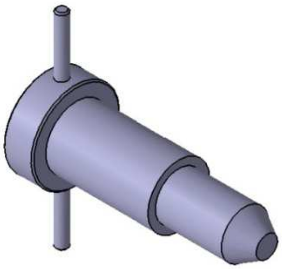


Figure 3: Bracket **Figure 4: Center Pin**

All the components designed for the Jig-Fixture assembly is done, keeping in mind the space constraints. Numerous permutations of dimensions were carried out and finalized on the basis of force analysis, cost and dimensional constraints.

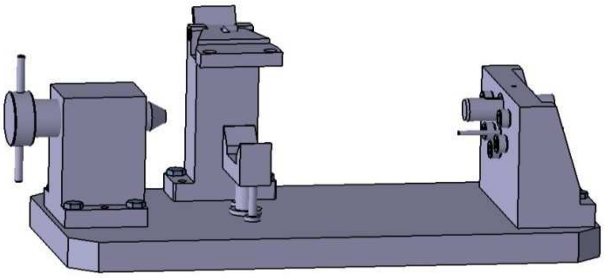


Figure 5: Final Design

FINITE ELEMENT ANALYSIS

Material Selection

Jig and fixtures are made from a variety of the materials. After studying properties, availability and cost of materials, we have come to the conclusion that MS is the most suitable material for jig and fixture. MS is selected because, it shows good resilience against deformation, and also it is more economical and easily available when compared to other materials like Cast Iron and Carbon Steel. MS can be easily hardened with methods like carburizing, case hardening, Shot peening etc. Properties of MS are given below in the table.

Table 1: Mechanical Properties of Mild Steel

Sr. No	Property	Value
1.	Ultimate Tensile strength	440 MPa
2.	Yield strength	370 MPa
3.	Modulus of Elasticity	210 GPa
4.	Density	7860 kg/m^3
5.	Poisson's Ratio	0.290
6.	Hardness	131 BHN

Meshing And Mesh Quality Check

Very first step in Analysis is meshing, which means discretization. The component is divided into finite number of small elements. This helps in converting the problem with infinite DOF's into finite DOF's. With the use of Hypermesh 13.0, meshing operation is done on Bracket and Center Pin. Dimensions of the model i.e. length(515mm), breadth(250mm) and height(200mm) are compatible, also the stresses are induced in all the 3 dimensions. Hence, 3-Dimensional solid meshing is done and parabolic tetra elements are used. Firstly, 2-Dimensional meshing is performed on the surface and then they are converted into 3-Dimensional tetra elements.

Primary mesh size is assumed as the smallest dimension of the object, and iterations were conducted with changing primary mesh size. Mesh size is selected with the help of Convergence criteria. The values of stresses, displacement and time taken for solving with different mesh sizes has been tabulated, and the optimum mesh size of 2 mm is selected having less deviation in stress & deformation values than other mesh size's results, along with optimum solving time.

Table 2: Mesh Size and Quality Criteria Comparison

Mesh Size (mm)	Number of Elements	Number of Nodes	Warpage	Aspect Ratio	Skewness	Jacobian
0.4	1532201	1943310	0.010	1.57	11	0.982
1	308742	386459	0.012	1.83	14	0.953
2	63121	77899	0.037	2.08	17	0.925
3	23583	28665	0.504	4.95	24	0.848
4	11849	14183	1.425	7.82	31	0.652

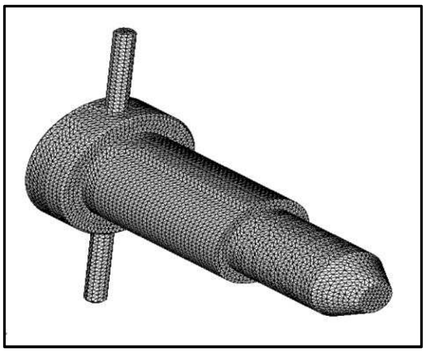


Figure 6: Meshed Center Pin

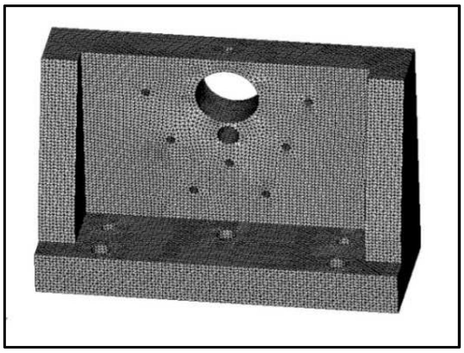


Figure 7: Meshed Bracket

Boundary Conditions

Application of loads on intended portions and fixing the bolted and fabricated areas is known as applying 'Boundary Conditions'. This is the most important step after meshing, because the results depend on the value and position of loads and constraints. Forces are evenly distributed over the nodes in the area, where the force is acting in reality.

For Center pin, we have constrained the threaded portion and applied the hand force of 150N on the handle, a vertically downward force of 74.4 N which is the self-weight reaction force of pinion on the front taper portion. For bracket, the bolting positions are constrained, and a force of 3kN which is the maximum load bearing capacity of thread is applied on rest buttons.

Hand force = 75N on each end.

Weight of Pinion = 180N

Vertical reaction Force of Pinion = 74.4N

Horizontal force acting on Bracket = 3kN

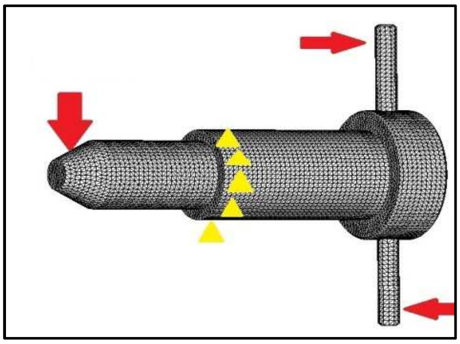


Figure 8: Boundary Conditions on Center Pin

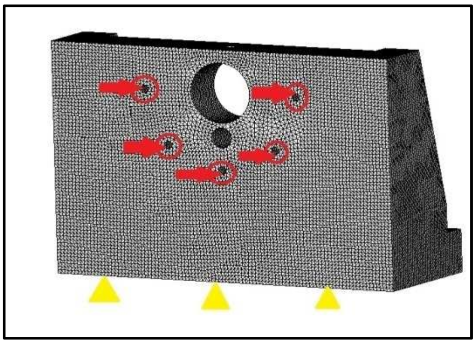


Figure 9: Boundary Conditions on Bracket

FEA Analysis

The model is solved in Optistruct solver of Hypermesh 13.0 using applied boundary conditions. After solving the problem, next step is known as post processing. In post processing, the results are shown in user understandable format i.e. Stress induced at the object is shown in the form of color bands, which shows how uniformly the stress is distributed and at which point, the magnitude of stress or deformation is maximum that shows where the possible failure can occur.

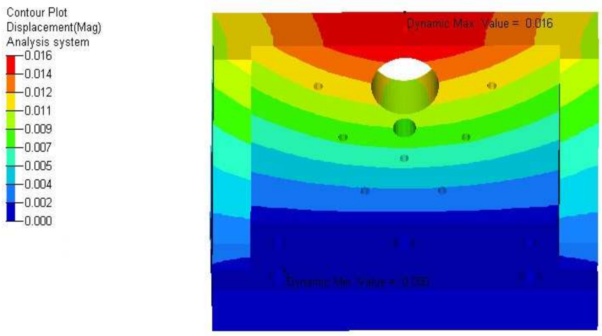


Figure 10: Deformation in Bracket

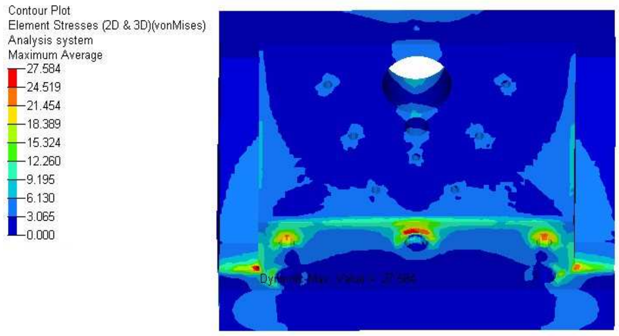


Figure 11: Stresses in Bracket

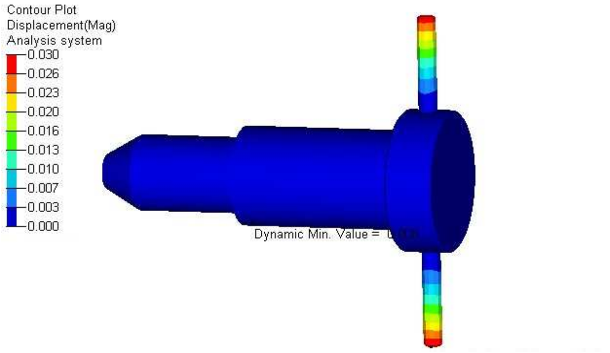


Figure 12: Displacement In Center Pin

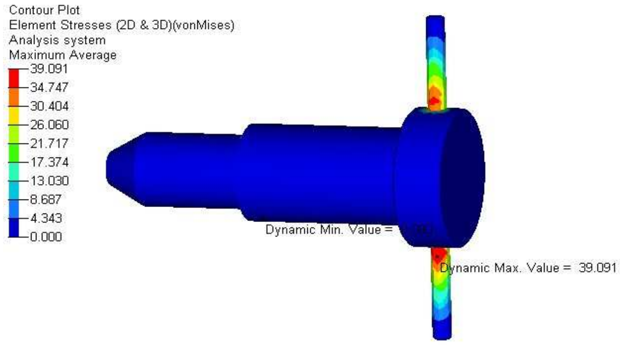


Figure 13: Stresses In Center Pin

Table 3: Resultant Stresses Induced in the Various Components of Jig and Fixture

Components	Induced Stress (N/mm^2)	Permissible Stress (N/mm^2)
Center pin	39	370
Fouling pin	32	370
Locating pin	23	370
Bracket	28	370
Jig plate	21	370

RESULTS AND CONCLUSIONS

The results achieved from CAE analysis are compared with the calculated results for validation purpose. CAE results are little bit deviated from the actual results, due to various parameters as shown below in the table.

Table 4: Comparison of CAE and Calculated Values

Sr. No.	Component	Stress (N/mm^2)		Deflection (mm)	
		CAE	Permissible	CAE	Permissible
1	Bracket	27.58	370	0.016	2
2	Center Pin	39.09	370	0.030	2

After analyzing the critical components in our model, we came to know that the values of stresses induced are less than the yield strength of the material, and also the values of deformation are in the range of few microns, which indicates that it will not affect the accuracy in the Drilling operation and can fulfill the purpose of precisely drilling the holes in the pinion. Simulation of impact of load and its consequences are observed using the Hypermesh 13.0. The obtained results were well within the limits according to safety and failure aspects. Therefore, it can be said that the design is safe and fool proof.

REFERENCES

1. *Charles Chikwendu Okpala, Okechukwu C. Ezeanyim (2015). The Design and Need for Jigs and Fixtures in Manufacturing, Science Research. Vol. 3, No. 4, 2015, PP. 213-219. DOI: 10.11648/j.sr.20150304.19*

2. *NBV Lakshmi Kumari and G. Prasanna Kumar (2015). Design and Analysis of Indexing type of drill Jig, IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE). Volume 12, Issue 2, PP. 46-51, DOI: 10.9790/1684-12214651*

3. *Deulgaonkar, V. R., (2018). Finite Element Analysis and Experimental Simulation of Chassis Mounted Platform for Off-Road Wheeled Combat and Transport Utility Vehicles.* International Journal of Vehicle Structures and Systems, 10*(1) 66-72. http://dx.doi.org/10.4273/ijvss.10.1.14*

4. *Khandare, N. H., & Deshmukh, S. An Elimination Type Of Pokayoke-A Game Changer Tool In The Propeller Shaft Assembly.*

5. *Deulgaonkar, V.R., (2016). Gradient Load Evaluation of Chassis Frame Mounted Specialised Structure Designed for Heavy Off-road Vehicles. International Journal of Vehicle Structures and Systems 8(2). 86-90. http://dx.doi.org/10.4273/ijvss.8.2.05*

6. *Deulgaonkar, V.R., (2016). Vibration Measurement and Spectral Analysis of Chassis Frame Mounted Structure for Off-road Wheeled Heavy Vehicles. International Journal of Vehicle Structures and Systems 8(2). 23-27.*

   http://dx.doi.org/10.4273/ijvss.8.1.05

7. *Reddy, P. R., & Saikiran, M. (2016). Aerodynamic Analysis of Return Channel Vanes in Centrifugal Compressors. International Journal of Mechanical Engineering (IJME), 5(1), 73-82.*

8. *Deulgaonkar, V. R. et al, (2015). Design Evaluation of Chassis Mounted Platform for Off-Road Wheeled Heavy Vehicles. International Journal of Vehicle Structures and Systems.7 (4). 100-106. http://dx.doi.org/10.4273/ijvss.7.3.03*

9. *Kollek, W. A. C. Ł. A. W., & Radziwanowska, U. R. S. Z. U. L. A. (2014). The modernization of gear micropump casing with the use of finite element method. International Journal of Research in Engineering & Technology, ISSN (E), 2321-8843.*

10. *Deulgaonkar, V. R. et al, (2014). Development and Validation of Chassis Mounted Platform design for Heavy Vehicles. International Journal of Vehicle Structures and Systems.6 (3). 51-57. http://dx.doi.org/10.4273/ijvss.6.3.02*